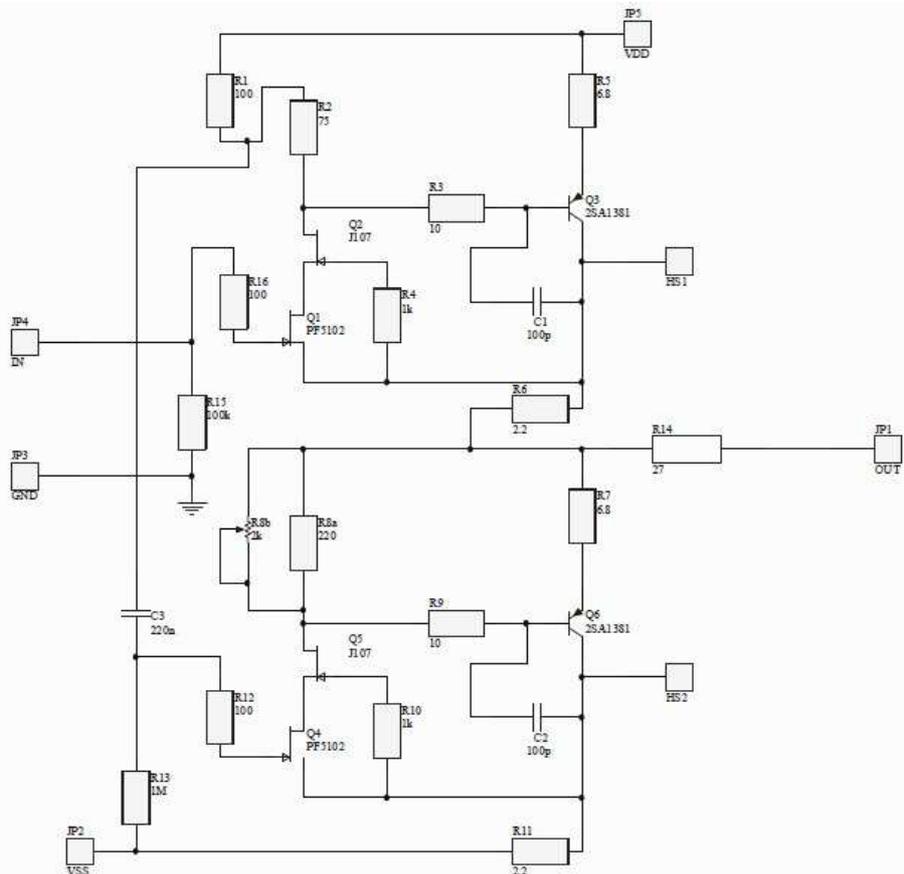


# Calvin Buffer for Paradise phono stage

Many thanks to „Calvin“ a.k.a. Christoph for letting us use his clever buffer design.

This is the assembly instructions for the Calvin Buffer, an add-on for the Paradise phono stage. But it can also be used as a standalone buffer. The schematic is shown below:

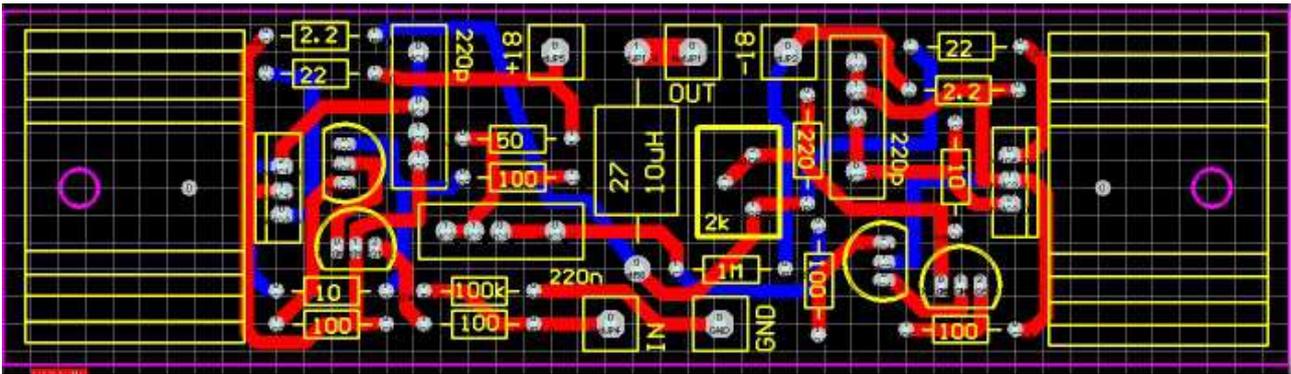


The JFET Q1 is cascaded with Q2, and its current flows through the base of Q3. R5 and R6 are used to adjust the idle current. R1 and R2 serve as the load resistance for Q1, and will provide a scaled-down version of the input voltage. The lower half of the buffer is identical to the upper half, except the trimpot that is used to adjust the idle current in the lower half, to allow setting the voltage offset to zero. And, the capacitor C3 will connect the scaled-down input voltage to the lower buffer circuit, to help with distortion cancellation. Not shown in the schematic is an inductor of approx. 10 $\mu$ H that is connected in parallel with R14.

The supply voltage is +/-18V, and the supply current is 30mA. The bandwidth is about 10MHz -3dB, and C1 and C2 can be used to adjust the square wave response for no overshoot.

The following picture shows the stuffing of the PCB. It is 120mm wide and fits exactly on top of the Paradise phono stage PCB. To the left and the right, the heatsinks with Q3 and Q6 are located. In the middle of the PCB, the output resistor R14 and inductor are located. The left half contains the upper buffer circuit, the right half contains the lower buffer circuit. Below are the „IN“ and „GND“ terminals, that can be connected with small wires to the „RIAA EXT“ connector on the Paradise PCB. Above are the „+18“ and „-18“ connectors as

well as „OUT“ that can be connected easily to the Paradise PCB as well.



### Bill of material

2	2SA1381	PNP power transistor
2	PF5102	n-JFET, IDSS=6mA, VTH = -1V
2	J107	n-JFET, IDSS=100mA, VTH=-3V
2	2.2Ohm	Resistor 0.5W
2	22Ohm	Resistor 0.5W
1	51Ohm	Resistor 0.5W
5	100Ohm	Resistor 0.5W
1	100kOhm	Resistor 0.5W
1	1MOhm	Resistor 0.5W
1	27Ohm	Resistor 0.5W
2	100Ohm	Resistor 0.5W
1	220Ohm	Resistor 0.5W
1	10uH	Inductor (see below)
1	2k	Trimpot 10 Turns, adjust from top
2	220pF	Foilcap 63V
1	220nF	Foilcap 63V
2	SK09	Heatsink Fischer, length 20mm
2	M3x10	mounting screw
2	M3x15	threaded bolt (nylon)

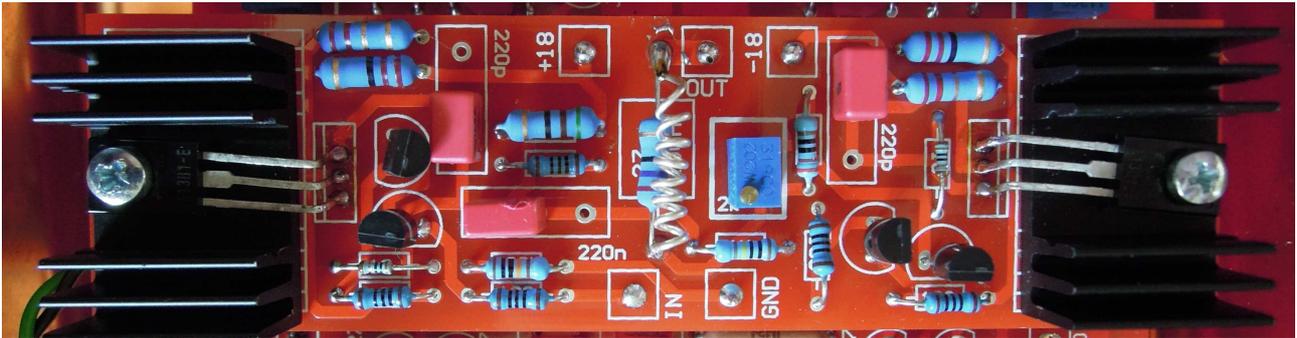
For the input transistors, also these can be used:  
2SK170 (IDSS=8...8.5mA) / cascode: PN4391

For the output transistors, a lot of choice is possible:  
BD 140 (Suffix -16) (Ccomp 180-220pF)  
MJE/MJD 350 (Ccomp 100-47pF)  
2SB 649/A (Suffix C oder D)  
2SA 1930 (Ccomp 180-220pF)  
MJE 253 (Ccomp 47-22pF)

### Assembly of the PCB

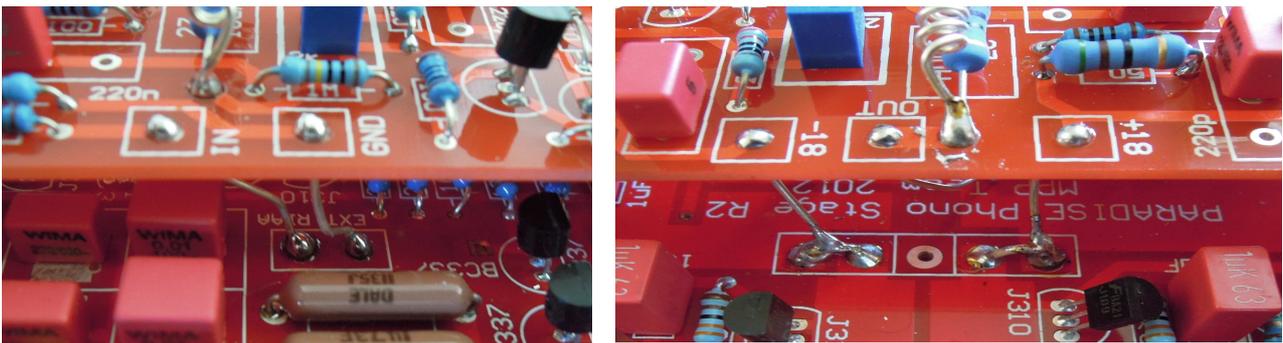
The assembly of the components is straightforward. Just start with the resistors, then the capacitors and trimpot, the transistors (pinout on the PCB is from left: Drain – Source – Gate, looking on the flat side of the package). The power transistors need to be mounted with the writing on top (pinout is from left: Emitter – Collector – Base, looking from top). First, drill a hole in the heatsink, bend the leads of the transistor in the right position with a flat plier, then tighten the heatsink together with the transistor to the PCB, and solder the

connections only afterwards. The inductor is a 10cm piece of wire, make 10 turns around a small screwdriver or similar, and take care that the windings are not touching each other. Simply solder this on top of the output resistor.



### Mounting onto the Paradise PCB

First, remove Q99 and Q91 from the original buffer. This deactivates it, no need to remove the rest, but you can if you want to. The new buffer will connect to the „RIAA EXT“ connector, the „+18“ and „-18“ jumpers at the shunt, and the 4.7ohm resistor from the original output buffer (output side). You may want to solder 5cm of some flexible cable to each of these points now.



When you used 15mm threaded bolts, the PCB should fit nicely on top of the Paradise PCB. The suggestion is to use Super-Glue to glue the bolts to the Paradise PCB for mechanical fixing. Then solder the cables to the right places, in case there should be trouble with the buffer, it can later be removed by opening the screws and desoldering the cables.

The buffer will use about 30mA per supply voltage, and the shunt regulator has absolutely no trouble with this. The rule of thumb however is that the shunt should consume about as much current as the load, so the current source of the regulator should deliver a touch more current. This is not strictly required, as said (and it works just perfect in my system), but for the perfectionists out there, you may want to reduce the 10Ohm power resistor in the current source to 8.2Ohm.

### Adjustments

The new buffer has adjustable offset, and should be adjusted to zero. For this, simply connect a multimeter between the „IN“ and „OUT“ connections, and adjust the trimpot to zero. That's it folks!

**What can go wrong:**

Constant offset of 60mV, no adjustment possible – the power bipolars are inserted the wrong way

Shunt regulator LEDs do not light up – check for short circuits on the buffer or the connection

Wandering offset, thermally very sensible – check the input ground connection (both „RIAA EXT“ connections must be connected to the buffer PCB)

Random offset, no adjustment possible: Check the output connection, it must go back to the main PCB, otherwise the Servo does not work

Please check [diyaudio.com](http://diyaudio.com), section „Analog Source“, thread „Paradise Builders“ for news and updates. This is also a perfect place to ask questions.